FINITE ELEMENT ANALYSIS

I. Introduction

The finite element method is a technique for obtaining approximate solutions to ordinary or partial differential equations. It finds application in situations where either the equation to be solved is complex or the geometry on which the solution is desired is complex. The method exchanges solving a single O. D. E. or P. D. E. on a large complex domain, for solving a much larger number of algebraic equations on smaller simply shaped pieces that make up the actual solution domain.

II. Solution Process

The process for solving a problem using the finite element method involves six major steps:

Step 1. Establish Governing Equations and Boundary Conditions.

In order to generate a valid approximate solution to a problem, the differential equation that governs the behavior and the corresponding boundary conditions for the problem must be determined. Once this is done the appropriate finite element formulation can be used to generate the solution.

Step 2. Divide Solution Domain into Elements.

In this step the entire solution domain is subdivided into “small” elements. Care is taken to make sure that enough elements are included to capture the behavior of the solution over the entire domain. Areas of particular interest and care are locations where critical values are expected, locations with large gradients, locations where the geometry changes suddenly, locations where boundary conditions and loads are applied. Typically, the larger the number of elements the better the approximation of the solution to the differential equation.

Step 3. Determine Element Equations.

Once the elements are formed, the algebraic equations to be solved are developed for each individual element. The form of the algebraic equations for every element will be the same. Differences from one element to the next will be due to changes in element size and properties. This is the power of the finite element method, the equations can be written once for a general element then they only need to be modified to reflect a particular elements geometry and properties.

Step 4. Assemble Global Equations.

Once all the element equations are generated they are put together to form a system of equations for the entire solution domain.

Step 5. Solution of Global Equations.

This system of equations is solved for the value of the dependent variable in the original differential equation at discreet points throughout the solution domain. Depending on the problem type there may be hundreds, thousands, tens of thousands, or even hundreds of thousands of points at which the solution to the differential equation is approximated.

Step 6. Solution Verification.

The accuracy of the solution must be verified before the results can be considered valid. One way to do this is to refine the mesh (increase the number of elements) and rerun the solution. If the value of the dependent variable at the discreet points in the mesh does not change significantly as the mesh is refined, the solution is deemed to be accurate.

III. Illustrative Example

As a means of illustrating the finite element method, the analysis of an axially loaded bar will be considered. Figure 1 shows an axisymmetric bar whose diameter varies linearly from one end to the other. The bar is fixed at the left end and loaded at the right end with a 500lbf. It is desired to know the displacement and stress distribution along the length of the bar.

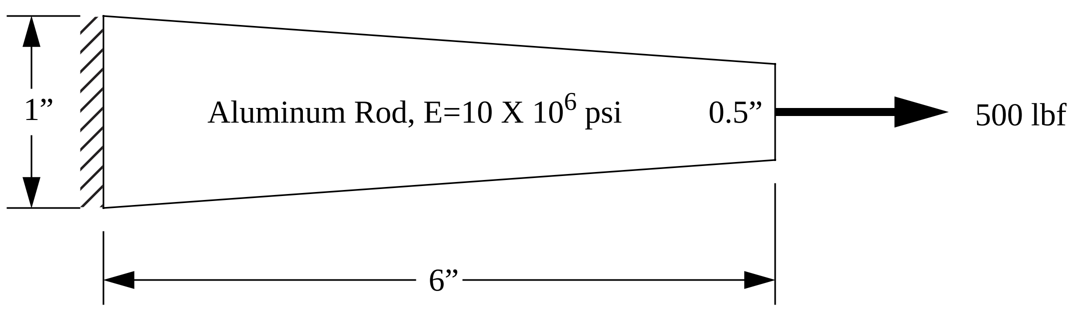


Figure 1 Axially Loaded Rod

Step 1. Establish Governing Equations and Boundary Conditions.

The equations describing the strain, stress, and reaction force within the bar are:

$$\varepsilon = \frac{du}{dx} \tag{1}$$

$$\sigma = E\varepsilon \quad \Rightarrow \quad \sigma = E\frac{du}{dx} \tag{2}$$

$$R = EA\frac{du}{dx}\ . \tag{3}$$

For this particular problem *E* will be constant, but *A* will be a function of x given by:

$$D = 1 - \frac{0.5}{6}x \qquad \Rightarrow \qquad A = \frac{\pi}{4}\left(1 - \frac{0.5}{6}x\right)^2 . \tag{4}$$

The boundary condition becomes $u = 0$ at $x=0$.

Having established the governing equation and the boundary condition, the development of an appropriate finite element formulation can be addressed. The element formulation is established by considering a small section of the rod and considering what forces are developed in the section as a result of displacements at each end of the section. The process is handled in three steps that are illustrated in figure 2.

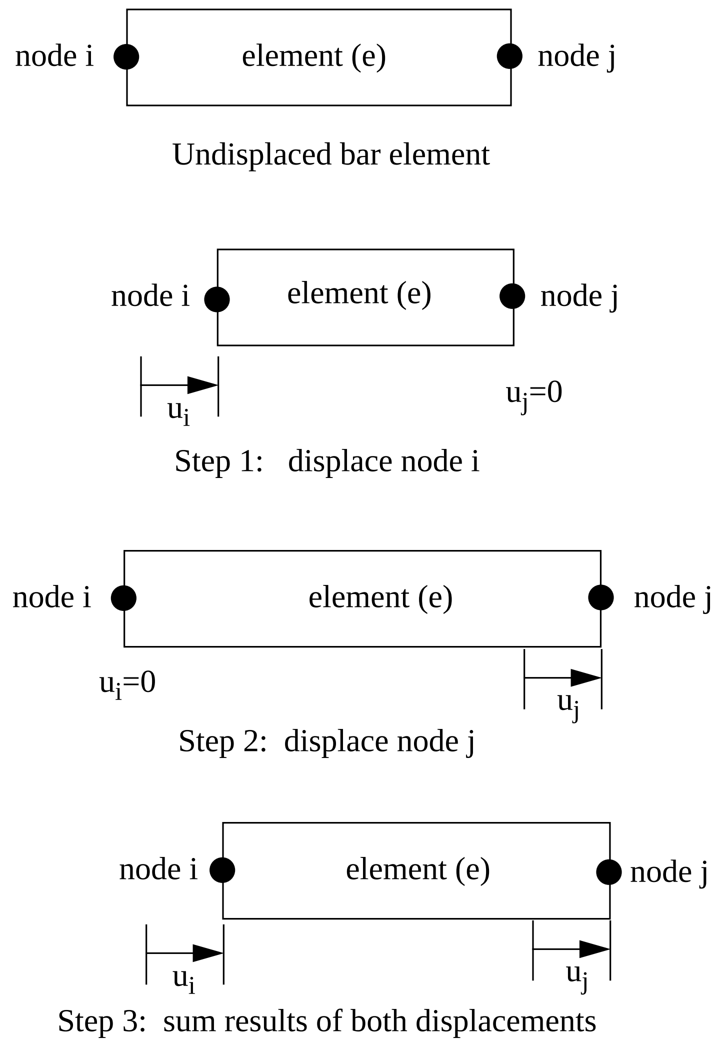


Figure 2 Determination of Element Formulation

In the first step the internal reactions for element (e) at nodes i and j are determined for a displacement at node i. The reaction at node i is:

$$R_{ii} = k_{ii}u_i \ , \tag{5}$$

where

$$k_{ii} = \left(\frac{AE}{L}\right)_{(e)} \ . \tag{6}$$

This is the force required to cause the displacement u_i. In order for the bar element to be in equilibrium there must be a corresponding reaction at node j, which is given by:

$$\sum F_x = R_{ii} + R_{ji} = 0 \ . \tag{7}$$

This becomes:

$$R_{ji} = -R_{ii} = -\left(\frac{AE}{L}\right)_{(e)} u_i \ . \tag{8}$$

Before going on to step two an explanation of the subscripting is in order. The first part of the double subscript designates the node at which the internal reaction is being calculated. The second part tells at which node the displacement is being applied. For example R_{ii} is the internal reaction at node i due to a displacement at node i, likewise R_{ji} is the internal reaction at node j due to a displacement at node i.

The second step determines the internal reactions caused by a displacement at node j:

$$R_{jj} = \left(\frac{AE}{L}\right)_{(e)} u_j \ , \tag{9}$$

and

$$\sum F_x = R_{jj} + R_{ij} = 0 \ , \tag{10}$$

and

$$R_{ij} = -R_{jj} = -\left(\frac{AE}{L}\right)_{(e)} u_j \ . \tag{11}$$

At this point the results from the previous two steps can be superimposed to give the internal reactions that result from arbitrary displacements at either end of the element:

$$R_i = R_{ii} + R_{ij} = \left(\frac{AE}{L}\right)_{(e)} u_i - \left(\frac{AE}{L}\right)_{(e)} u_j \ , \tag{12}$$

and

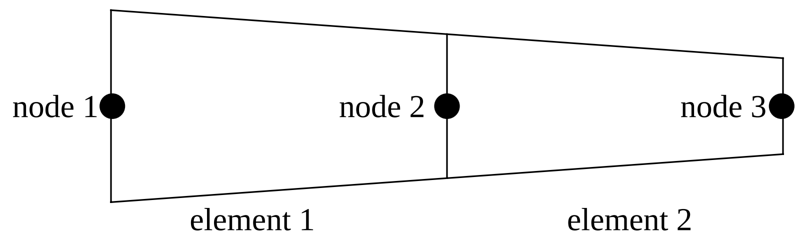
$$R_j = R_{ji} + R_{jj} = -\left(\frac{AE}{L}\right)_{(e)} u_i + \left(\frac{AE}{L}\right)_{(e)} u_j \ . \tag{13}$$

Equations 12 and 13 are the algebraic approximation to the differential equation describing the internal forces within the bar, that was given by equation 3. Equations 12 and 13 can be combined in matrix form to represent the contribution of a general element to the overall solution:

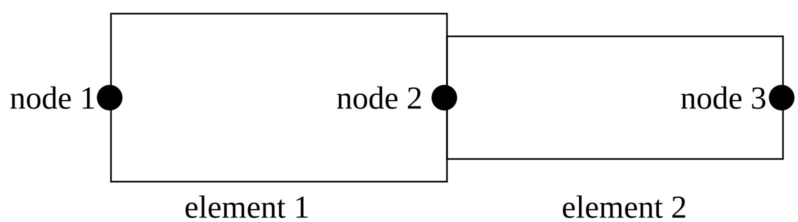
$$\begin{Bmatrix} R_i \\ R_j \end{Bmatrix} = \begin{bmatrix} \left(\frac{AE}{L}\right) & -\left(\frac{AE}{L}\right) \\ -\left(\frac{AE}{L}\right) & \left(\frac{AE}{L}\right) \end{bmatrix}_{(e)} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} . \tag{14}$$

The 2X2 matrix in the equation above is referred to as the element stiffness matrix. This is because it relates the force in the element to the displacements in the element in a manner similar to the stiffness in a linear spring. It should be noted that to simplify the formulation of the general element equation, the properties for an individual element are assumed to be constant for a given element. They may vary from one element to the next. For equation 14 average values of A, and E would be used for an element.

Having completed the first step of the finite element process, the second step of dividing the solution domain into elements can be done. For this example two elements will be used. This subdivision is shown in figure 3.



Element Subdivision



Constant Property Approximation

Figure 3 Finite Element Division

The element properties become:

$$E_{(1)} = 10 \text{ X } 10^6 \text{ psi}$$
$$D_{(1)} = 0.875''$$
$$A_{(1)} = 0.601 \text{ in}^2$$

$$E_{(2)} = 10 \text{ X } 10^6 \text{ psi}$$
$$D_{(2)} = 0.625''$$
$$A_{(2)} = 0.307 \text{ in}^2.$$

Determining the stiffness matrices for all the elements in the problem is the third step in the process. For this example there are two elements and the quantity $\left(\frac{AE}{L}\right)_{(e)}$ must be calculated for each. Based on the values given above the element stiffness matrices become:

$$[K]_{(1)} = \begin{bmatrix} 2 \times 10^6 & -2 \times 10^6 \\ -2 \times 10^6 & 2 \times 10^6 \end{bmatrix} \text{ lbf/in}$$

$$[K]_{(2)} = \begin{bmatrix} 1 \times 10^6 & -1 \times 10^6 \\ -1 \times 10^6 & 1 \times 10^6 \end{bmatrix} \text{ lbf/in}$$

The element equations become:

$$\begin{Bmatrix} R_1 \\ R_2 \end{Bmatrix}_{(1)} = \begin{bmatrix} 2 \times 10^6 & -2 \times 10^6 \\ -2 \times 10^6 & 2 \times 10^6 \end{bmatrix}_{(1)} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}_{(1)}$$

$$\begin{Bmatrix} R_2 \\ R_3 \end{Bmatrix}_{(2)} = \begin{bmatrix} 1 \times 10^6 & -1 \times 10^6 \\ -1 \times 10^6 & 1 \times 10^6 \end{bmatrix}_{(2)} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}_{(2)} .$$

Notice that both elements make a contribution to the internal reaction for node 2.

Step 4 is the assembly of the individual element equations into the system of equations that describes the entire solution domain. The force vector will contain three entries R_1, R_2, and R_3. The displacement vector will contain u_1, u_2, and u_3. The global stiffness matrix will be a 3X3 matrix that is formed by combining the 2X2 element stiffness matrices. The global system of equations is:

$$\begin{Bmatrix} R_1 \\ R_2 \\ R_3 \end{Bmatrix} = \begin{bmatrix} 2 \times 10^6 & -2 \times 10^6 & 0 \\ -2 \times 10^6 & 2 \times 10^6 + 1 \times 10^6 & -1 \times 10^6 \\ 0 & -1 \times 10^6 & 1 \times 10^6 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} .$$

Once the global equations have been assembled the boundary conditions and any other known values can be incorporated into the global system. For this example the displacement at the left end is u_1=0, the internal reaction at the right end equals the externally applied force of 500 lbf, and R_2=0 because there is no externally applied force at node 2. The global equations become:

$$\begin{Bmatrix} R_1 \\ 0 \\ 500 \end{Bmatrix} = \begin{bmatrix} 2 \times 10^6 & -2 \times 10^6 & 0 \\ -2 \times 10^6 & 2 \times 10^6 + 1 \times 10^6 & -1 \times 10^6 \\ 0 & -1 \times 10^6 & 1 \times 10^6 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix} ,$$

where the remaining unknowns are the displacements at nodes 2 and 3 and the reaction force at node 1

The solution to this system of equations is:

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 2.5 \times 10^{-4} \\ 7.5 \times 10^{-4} \end{Bmatrix} \text{ in } .$$

The exact solution to this problem is determined by solving equation 3 using equation 4, and using the fact that the reaction at any location within the bar is constant at 500 lbf. The result is:

$$u(x) = 7.64 \times 10^{-4} \left[\frac{1}{\left(1 - \frac{0.5}{6}x\right)} - 1 \right] \text{in}$$

Table 1 compares the finite element solution and the exact solution for the two element case.

Table 1: Comparison of Finite Element and exact Solutions

x (in)	$u_{F.E.}$ (in)	u_{exact} (in)	% Error
0	0	0	0
3	2.5E-4	2.55E-4	2
6	7.4E-4	7.64E-4	3

The final step in the finite element solution process is to verify the solution. This is done by refining the mesh, i.e. adding more elements, and examining the effect on the solution. If there is no significant change in the solution with the addition of more elements, then the solution is considered adequate. Figure 4 shows the comparison between the exact solution and the finite element solution for meshes with 2, 4, and 8 elements.

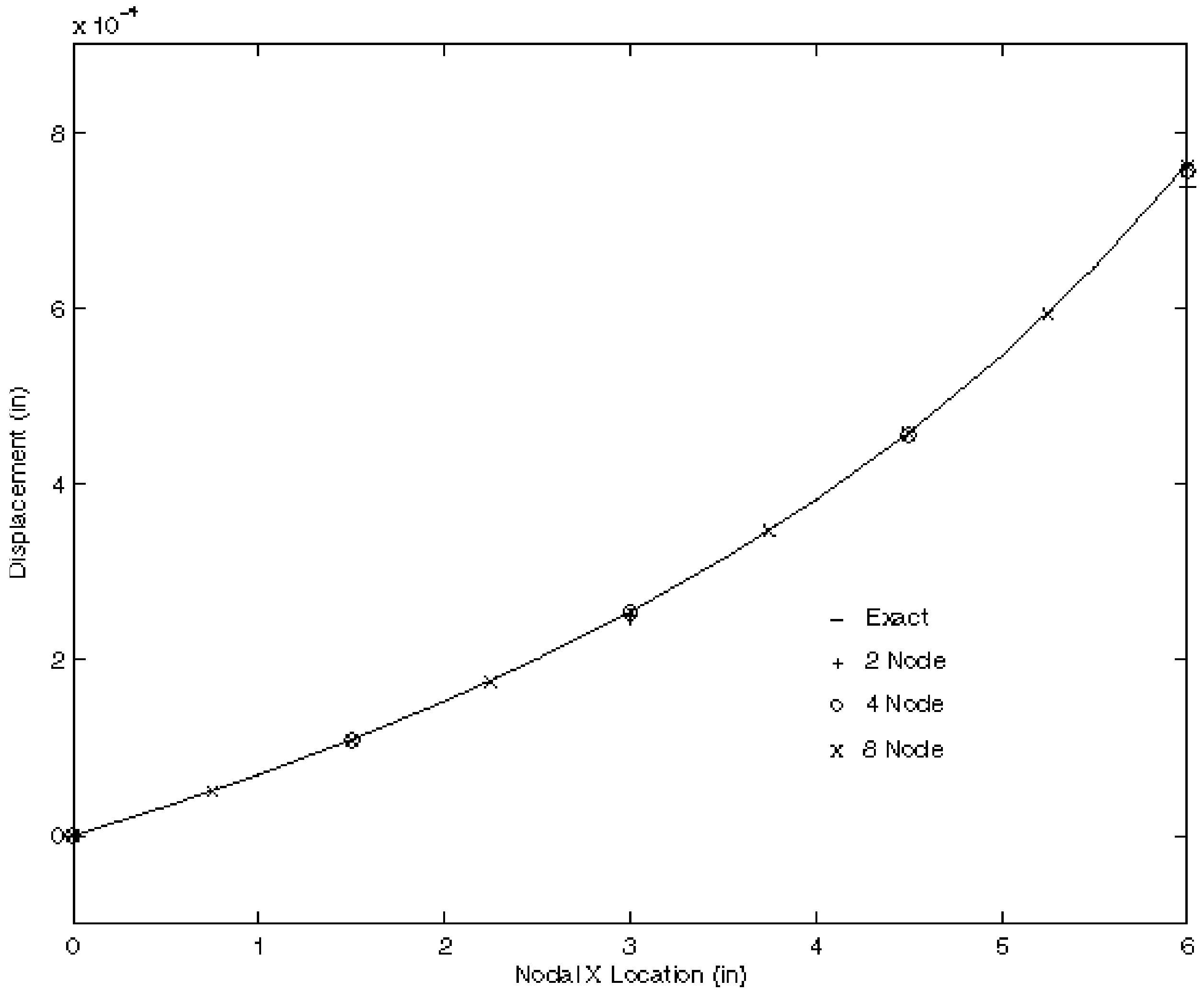


Figure 4 Finite Element Results

Finite Element Analysis Homework

An axisymmetric rod, shown below, has a displacement of 2.0×10^{-4} in at the right end. The left end of the rod is fixed. Determine the displacement, strain, stress, and force in the rod. Divide the rod into three equal length elements. Recall that the strain, stress, and force are given by:

$$\varepsilon = \frac{du}{dx} = \frac{\Delta u}{\Delta x} \tag{1}$$

$$\sigma = E\varepsilon \quad \Rightarrow \quad \sigma = E\frac{du}{dx} \tag{2}$$

$$R = EA\frac{du}{dx} \ . \tag{3}$$

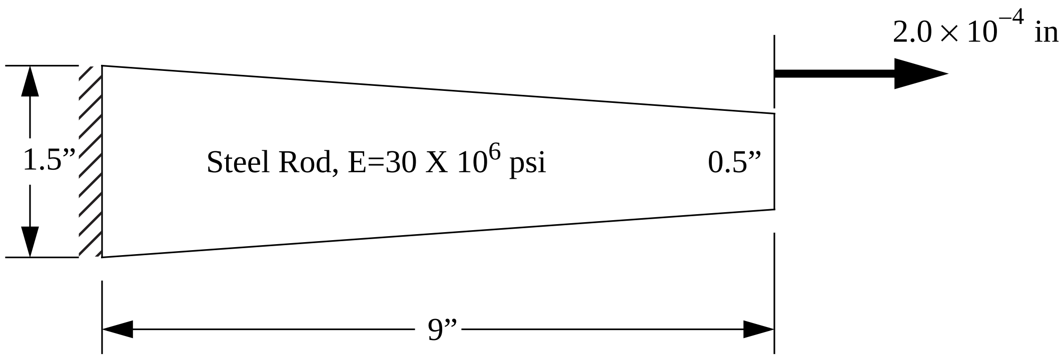


Figure 1 Axially Displaced Rod

Compare the results for displacement and stress to the exact solution, which is determined from the governing equation:

$$\frac{d}{dx}\left(AE\frac{du}{dx}\right) = 0 \ .$$

Note: The finite element solution will give the displacements at the nodes, but the stress result based on equation 2 will be an average value for the element.